AD625566

A FLOW MODEL AND ANALYSIS OF A ROCKET PAIR IN CLOSE PROXIMITY

HUGH A. THOMPSON AND HENRY F. HRUBECKY

PREPARED FOR
THE ADVANCED SYSTEMS LABORATORY
U. S. ARMY MISSILE COMMAND
REDSTONE ARSENAL, ALABAMA

Department
of
Mechanical Engineering
Tulane University

January 1965

A FLOW MODEL AND ANALYSIS OF A ROCKET PAIR IN CLOSE PROXIMITY

JANUARY, 1965

HUGH A. THOMPSON AND HENRY F. HRUBECKY

PREPARED FOR THE

ADVANCED SYSTEMS LABORATORY

U. S. ARMY MISSILE COMMAND

REDSTONE ARSENAL, ALABAMA

BY

DEPARTMENT OF MECHANICAL ENGINEERING
TULANE UNIVERSITY
NEW ORLEANS, LOUISIANA

TABLE OF CONTENTS

ABSTRACT

In the flight of a pair of rockets or the salvo firing of several rockets, the trailing ones may be deflected toward the trajectory of the leading rocket. The progress that has been made in investigating the aerodynamic aspects of this problem are presented in this report. Two areas in the flow field have been isolated as regions in which an aerodynamic interaction could occur between rockets. A mathematical model has been developed which simulates the simpler of these regions: the perturbation of an inviscid, incompressible flow field by the passage of the leading rocket and the subsequent flight through this disturbed flow by the trailing rocket. This model employs a numerical method known as the method of successive overrelaxations to compute the resultant of the aerodynamic pressure force and the point at which it is concentrated for both a leading and a trailing rocket. The data that are required by the model and the problem areas that remain in the preparation of these data are presented and discussed.

INTRODUCTION

If a pair of rockets are placed close together and aimed along parallel trajectories, it might be expected that the points of impact would be separated, at least on a statistical basis, by a distance equal to the initial spacing. This is not always the case. When rockets are fired essentially simultaneously or in closely spaced salvos, the trailing rockets move toward the flight path of the leading rockets. Apparently in extreme cases the trailing rocket may even cross the wake of the leading one.

The purpose of this study has been to isolate those mechanisms which would tend to deflect a trailing rocket. Further, having determined the possible origins of the interference to develop a mathematical model which could be used to determine rocket spacings or delay times that would assure rocket flights in pairs or salvos with accuracies comparable to those obtainable with single round firings.

This particular problem of decreased accuracy is associated with essentially simultaneous firings of groups of two or more rockets located in close proximity to one another. This association suggests strongly that the deflection of the trailing rocket is due to an aerodynamic interaction. Such possible sources of inaccuracy as wind gusts during the time of flight, mal aim and thrust mal alignment would not be confined to the salvo firing situation, nor preferential in their action toward the trailing rocket. These factors should on a statistical basis disrupt the flight of the leading rocket with a frequency comparable to that encountered with the trailing one. Deformation of the launching platform of the trailing rocket under the impact of the exhaust of the lead rocket has not been pursued as a possible source of inaccuracy. Although this particular interaction shows the requisite discrimination for the trailing rocket and could be a significant factor in salvo firings, it would be totally absent in essentially simultaneous firings from separate launching platforms. In this case the deformation effects would be closely equivalent for both rockets and yet the deflection of the trailing rocket still occurs.

An hypothesis attributing the observed deflection of trailing rockets to aerodynamic forces seems generally to fit the facts. The perturbing mechanism is operative at all times and during all firings. The magnitude of the interacting forces is a function both of the spacing between rockets and the deviation in time between firings. Finally, there exists a decidedly greater influence on the trailing rocket than on the leading one.

Examination of the flow field shows two major areas in which an aerodynamic interaction may occur. One is the region (1)* of turbulent decay of the jet exhausting from the leading rocket. In this region the potential core of the jet decays by turbulent mixing with the atmosphere and mean velocities change from the maximum values at jet exhaust to the free stream velocity existent in the undisturbed atmosphere at some distance from the rocket. Large velocity gradients occur in this area and a rocket trailing in this region would be subjected to substantial aerodynamic pressure differences. The direction of the resultant pressure forces acting on the trailing rocket would be toward the trajectory of the leading one. In order that this interaction occur, the trailing rocket must be located several rocket lengths behind its leader. During either

---

* Numbers in parentheses refer to the bibliography located at the end of the report.

simultaneous firings or salvc firing at time intervals of sufficient duration to allow the turbulence induced by the first rocket's jet wake to decay away, drift of the trailing rocket which is caused by this interaction would not occur.

An aerodynamic interaction clearly arises in the disruption of the flow field by the leading rocket and the subsequent flight of the second rocket through this disturbed environment. The region outside the jet wake of the leading rocket is disturbed by the passage of the body of the rocket. Fluid is accelerated away from and to the rear of the body. Velocity gradients, although of considerably smaller magnitude than in the case of the turbulent mixing region of the exhausting jet, are induced in the vicinity of the leading rocket. The flight of the trailing rocket in this second disturbed flow region also results in aerodynamic pressure forces tending to draw the two rockets together. This interaction occurs even when the distance one rocket lags behind the other is zero. The disturbance of the atmosphere which is created by the passage of the lead rocket spreads radially outward from the longitudinal axis of the rocket and the distance to the outermost extent of this disturbance increases with the distance traversed from the nose. Due to this widening cone of disrupted flow, the interacting forces will increase as the trailing rocket falls progressively farther behind. The interactions arising in this region are then also preferential toward the trailing rocket.

In either area, the center of pressure on the trailing rocket, that is, that point at which the aerodynamic forces may be considered to be concentrated, will probably not coincide with the location of its center of mass. The overturning moment resulting from such a force distribution tends to cause a yawing of the trailing rocket. This tendency towards a rotational motion about its center of mass is in addition to the translational drift toward the trajectory of the leading rocket. Even though both spin of a portion of the rocket about its longitudinal axis and fins on the aft sections are employed to minimize deviations from a zero angle of attack (and assure nose first impact), yaw of the rocket prior to burnout will bring some component of the thrust to bear as a deflecting force. If interacting forces maintain a preferred direction of yaw, the magnitude of the force deflecting the trailing rocket will be greater than the forces due to aerodynamic loadings alone. One of the objectives of this study could, then, be restated as the isolation of those mechanisms which result in a favored orientation in the yaw of the trailing rocket such that a component of thrust deflects it toward the path of the leading rocket.

This report summarizes the progress that has been made toward the development of a mathematical model of the interaction occurring between two rockets through the simpler of the two mechanisms, the disturbance of the flow field by the passage of the leading rocket. Physically this amounts to treatment of the case of essentially simultaneous firings in which no part of the jet exhaust from the lead rocket impinges upon the trailing rocket. By a number of simplifications and assumptions, this case has been brought within the realm of accessible solution. Indeed, such may also be feasible in the second region; however, this region is beset with a number of problems that are not present in the simpler case. The turbulent interchange of momentum and heat combined with the mixing that occurs between the products of combustion and the atmosphere

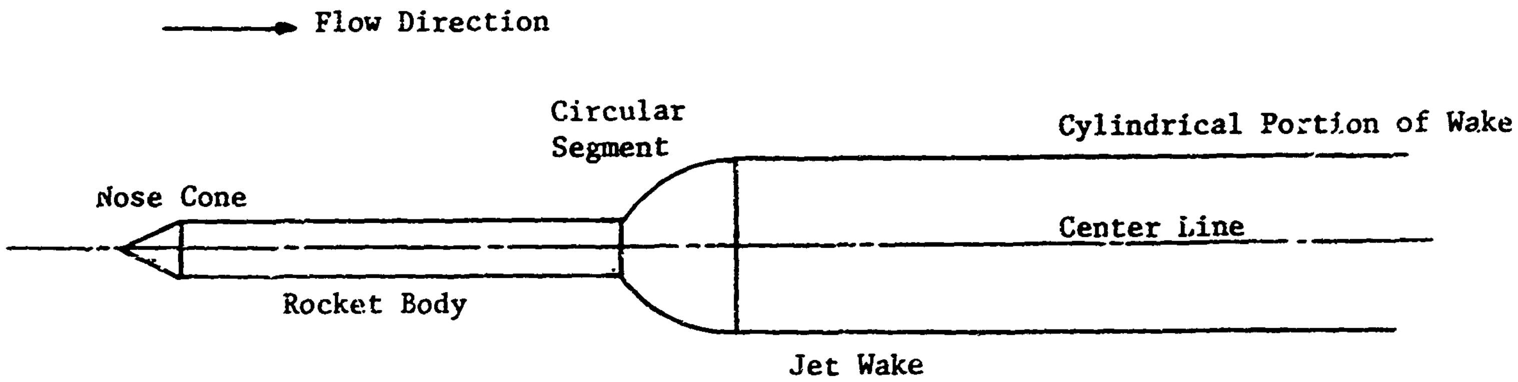

FIGURE 1. SIMPLIFIED ROCKET GEOMETRY

boundary of the exhausting jet is greatly simplified. The mixing processes through which the products of combustion intermingle with the atmosphere have been completely eliminated by visualizing the jet as a solid body. In cross section the boundary between the interior of the jet and the atmosphere consists of a circular segment attached to the rear of the body of the rocket and extending rearward until its tangent is parallel with the direction of flow. Beyond this point the remaining portion of the wake is simulated by a cylinder extending to infinity.

The conical tip was used in preference to a more realistic ogive because of its simpler mathematical form. If the hypotheses advanced earlier are verified, an ogive-type profile may be incorporated as a refinement on the present work.

Little information is available on the shape of the boundary of a jet exhausting into a flowing stream. The investigations of Love, Grigsby, Lee and Woodling (2) have shown that the boundary of an axisymmetric free jet exhausting into still air may for its initial length be represented by an arc of a circle. Curves are presented which show

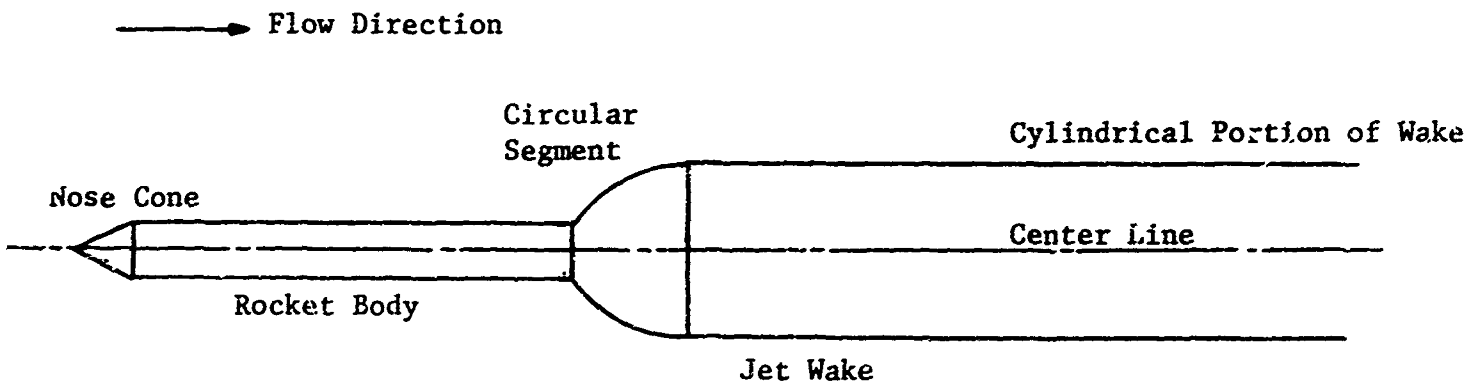

FIGURE 1. SIMPLIFIED ROCKET GEOMETRY

boundary of the exhausting jet is greatly simplified. The mixing processes through which the products of combustion intermingle with the atmosphere have been completely eliminated by visualizing the jet as a solid body. In cross section the boundary between the interior of the jet and the atmosphere consists of a circular segment attached to the rear of the body of the rocket and extending rearward until its tangent is parallel with the direction of flow. Beyond this point the remaining portion of the wake is simulated by a cylinder extending to infinity.

The conical tip was used in preference to a more realistic ogive because of its simpler mathematical form. If the hypotheses advanced earlier are verified, an ogive-type profile may be incorporated as a refinement on the present work.

Little information is available on the shape of the boundary of a jet exhausting into a flowing stream. The investigations of Love, Grigsby, Lee and Woodling (2) have shown that the boundary of an axisymmetric free jet exhausting into still air may for its initial length be represented by an arc of a circle. Curves are presented which show the radius of the arc and the maximum diameter of the jet as functions of the pressure ratio and divergence angle of the discharging nozzle. Although the free stream velocities associated with the flight of pairs of rockets are not zero, deviations from the situation where a jet exhausts into still air should not be significant, at least, insofar as this problem is concerned. Until more directly applicable data become available, the material of Love, et al, is recommended for use in determination of the initial curvature of the exhausting jet. That is, it is recommended that determination of the radius of the circular arc be made independent of the free stream velocity. The correlations presented by Love, et al, were developed from numerous solutions employing the method of characteristics. As this method does not account for the turbulent decay of the potential core of the jet, the results cannot be applied much beyond the point where the boundary of the jet turns parallel with the free stream flow; that is, the point at which the jet reaches its maximum diameter. It is at this point that the turbulent decay of the exhausting jet commences. This is the region that has been approximated by a cylinder extending to infinity. It is the least realistic of the areas postulated in the simulation of the rocket and its jet. Since this phase of the investigation is concerned primarily with disturbances which occur in proximity to the body of the rocket, the simulated wake is not considered a gross approximation unless the distance of trail of one rocket behind the other is great.

In the initial stages of a firing, where rocket velocities are less than 300 fps, the region outside the wake may be considered both inviscid and incompressible These conditions when combined with the assumption of steady state require that the stream function in this region must satisfy Laplace's equation. That portion of total time of flight of a rocket which is spent in the incompressible flow region comprises a disproportionately large percentage of the total. Further, the magnitudes of the forces deflecting the trailing rocket are very small when compared to the thrust and their action must be well established in the incompressible flow portion of the flight if a significant deviation is to occur. For these reasons the initial investigations were restricted to incompressible flows.

Under the conditions outlined above, the flow is called potential (3) and the techniques of conformal mapping are, in theory, applicable to describe the flow. However, our attempts to utilize the complex plane to obtain the potential flow field surrounding two rockets were unsuccessful. The Schwartz-Christoffel transformation will not admit the curved portion of the boundary of the exhausting jet. The method of Levi-Civita will admit curved boundaries, but there does not appear to be a way to map the flow field surrounding two rockets into a single region in the complex plane.

When attempts to obtain an analytical solution were not successful, a computer program employing numerical methods was developed to calculate the interacting forces and their centers of pressure. Of the various numerical techniques that were considered, the method of successive overrelaxations was selected as most promising in terms of both adaptability and probable minimum convergence time. This technique was applied to a five point finite difference analogue to Laplace's equation in which the stream function was the independent variable. The domain of definition of this variable consists of a rectangular field enclosing two simulated rockets. By means of the input data the size of the rockets, the distance between them and the distance of trail of one behind the other may be varied and the influence of these variables on the force of interaction examined. In addition to input data describing the geometric disposition of the rockets, the values of the stream function on the boundaries of the rectangle must also be specified and by changing these bounding values various rocket velocities may be examined.

This program has been compiled and checked by means of two example problems. The influence of many factors must be established before the data from this program may be considered reliable. However, on the basis of the fragmentary information from the two example problems it does appear that a sizable force of interaction is present and the hypothesized mechanism of the interaction may be valid.

DESCRIPTION OF INPUT DATA TO PROGRAM

Figure 2 shows a sketch of the rectangular flow field superimposed upon two rockets. Those quantities that specify the geometry are shown on the figure as dimensions. These quantities are input data to the program and may be varied, within certain broad limits, to determine their influence upon the aerodynamic interaction. The following tabulation presents a complete list of the data that must be provided in the preparation of a problem for machine computation. As it is not feasible to provide logic within the program that is capable of handling all possible configurations, certain restrictions must be placed on the variables and some combinations of variables. These restrictions are noted in the description of each variable and summarized in a subsequent table. The variable H which is frequently referred to in the following tabulation is the value of the mesh size; that is, the distance between adjacent nodal points in the relaxation network used in the method of successive overrelaxations. All distances in the program are measured in feet. The listing of the variables that specify the geometry of the rocket is as follows:

XL1 - The altitude of the cone representing the tip of the rocket.

XL2 - The length of the cylindrical portion of the rocket. The value of XL2/H must be an integer which is greater than or equal to 3.

R1 - The radius of the cylindrical portion of the body of the rocket. The value of XL2/H must <u>not</u> be an integer.

DELTA - The angle measured in radians that the tangent to the circular portion of the boundary of the exhausting jet makes with the horizontal at the point where the boundary joins the cylindrical portion of the wake.

RW - The radius of the cylindrical portion of the solid body that simulates the boundary of the exhausting jet. The ratio of RW/H must <u>not</u> be an integer.

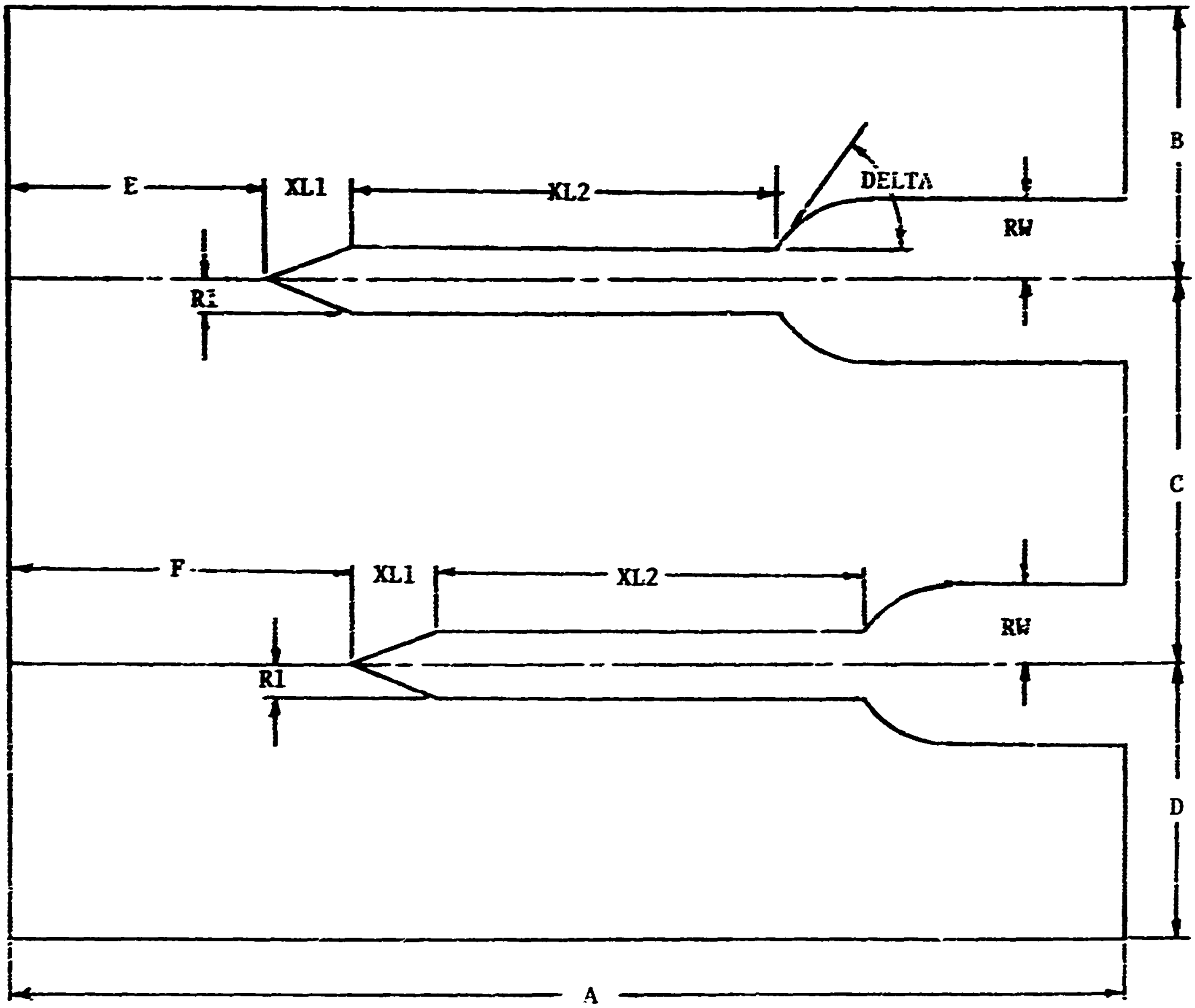

FIGURE 2. FLOW FIELD DIMENSIONS AND GEOMETRIC CONFIGURATION OF A PAIR OF ROCKETS SIMULATED IN THIS FIELD

The variables that specify the dimensions of the flow field and position the rockets in that field are:

IMAX - An integer determining the length of the flow field that lies parallel to the direction of flight. The dimension A indicated in the figure equals the product of H and IMAX minus one. IMAX must be less than or equal to 34.

B - The distance between the longitudinal center line of the leading rocket and that edge of the flow field which lies nearest to and parallel with its centerline. The value of B/H must be an integer which is greater than or equal to the value of 3.0 + (RW/H).

D - The distance between the longitudinal center line of the trailing rocket and that edge of the flow field which lies nearest to and parallel with the centerline. The value of D/H must be an integer which is greater than or equal to the value of 3.0 + (RW/H).

JMAX - An integer determining the width of the flow field, the dimension that lies perpendicular to the direction of flight. The dimension C indicated in the figure is defined by the equation, $C = H\cdot(JMAX - 1) - B - D$. The value of JMAX must be an integer which is less than 28.

E - The distance between the upstream edge of the flow field and the vertex of the nose cone of the leading rocket. The value of E/H must not be an integer and must exceed the value 2 . The value of (E + XL1)/H must, however, be an integer.

F - The distance between the upstream edge of the flow field and the vertex of the nose cone of the leading rocket. The value of F/H must not be an integer and must exceed the value 2. The value of (F + XL1)/H must, however, be an integer.

In most iterative numerical techniques, a network is superimposed over the flow field and the finite difference approximations to the controlling equations are repeatedly applied until they are closely satisfied at every point of intersection in the mesh. The nodal points employed in the square network of this program are identified by means of a matrix-type numbering system in which each point is assigned two integers I and J . In locating a particular point I, J, the value of I is the number of the column of intersecting mesh points in which it is found; the columns are numbered sequentially starting with the upstream boundary of the flow field as the first. The value of J is the number of the row in which the point is located. The rows are numbered sequentially downward with the upper horizontal boundary taken as the first. The value of the stream function at the nodal point I, J is given the symbol X(I,J) . Values of the stream function at each of the nodal points lying on the boundary of the flow field must be specified in the input data. By specifying the bounding values of the stream function, the velocity distribution at the boundary of the flow field is fixed and the boundary conditions for the problem are cast. Those data that must be specified in the input are:

X(1,1), X(1,2), ... , X(1, JMAX) - These JMAX values of the stream function at the nodal points on the upstream boundary determine the free stream conditions. The values of the stream function on the upper horizontal boundary are constant and equal to X(1,1) while those on the lower horizontal boundary are also constant, but equal to X(1, JMAX).

X(IMAX, 1), X(IMAX, 2), ... , X(IMAX, JMAX) - These are the stream function values on the downstream boundary to the flow field. As will be discussed later, the specification of these values is one of the most difficult problems associated with this program. It will occur that some of the nodal points on this boundary lie inside the solid boundary which represents the exhausting jet. Values of the stream function specified at these points are never used in the program. Although their value is unimportant, there must be JMAX values specified so that some arbitrary value must be assigned at these points.

XB1 - The value of the stream function at every point on the boundary of the leading rocket.

XB2 - The value of the stream function at every point on the boundary of the trailing rocket.

RHOI - The density in slugs per cubic foot of the undisturbed fluid through which the rockets pass.

Certain quantities must be specified in the input information which are associated solely with the computational method. These data are:

H - The distance between adjacent nodal points in the relaxation network.

TOL - The value of this variable when multiplied by one hundred is approximately the maximum percentage error that is acceptable in the computation of the stream function.

W - This term is known as the convergence factor. It is used in the method of successive overrelaxations to minimize the number of times the computation must be performed. The convergence factor must be optimized with respect to the dimensions of the flow field. When the input value is made zero, the program will approximate the optimum value according to the relation:

$$\bar{\mu} = 0.5 \left[\cos\left(\frac{\pi H}{A}\right) + \cos\left(\frac{\pi H}{B+C+D}\right)\right] \quad (1)$$

$$W = 1.0 + \left\{\frac{\bar{\mu}}{1.0 + \sqrt{1.0-\ ^{2}}}\right\}^{2} \quad (2)$$

If a nonzero value of W is read into the problem, then the internal computation of W will be omitted and the input value used instead.

TABLE I. SUMMARY OF RESTRICTIONS ON INPUT DATA

The following conditions must be satisfied by the input data:

For the individual variables

$XL1/H \neq n$*

$XL2/H = n \geq 3$

$R1/H \neq n$

$RW/H \neq n$

$IMAX = n \leq 34$

$B/H = n \geq 3.0 + (RW/H)$

$D/H = n \geq 3.0 + (RW/H)$

$JMAX = n \leq 28$

$2 < E/H \neq n$

$2 < F/H \neq n$

For the following combination of variables

$B/H - RW/H > 3$

$(E + XL1)/H = n$

$(F + XL1)/H = n$

---

* The symbol n as used in this tabulation denotes any positive integer.

The data which are normally obtained from this program are the resultant force and the center of pressure of that force on both the leading and the trailing rockets. The center of pressure is located by specifying the longitudinal distance between the vertex of the cone representing the tip of the rocket and that point on the body of the rocket at which the resultant force may be considered to be concentrated. The terminal values of the stream function at all nodal points in the flow field may also be obtained by properly positioning a sense switch on the computer console. Although the distribution of aerodynamic pressure forces on the body of the rocket is computed in determining the resultant force, it is not accessible as output data.

THEORETICAL DISCUSSION

Under the assumed conditions of inviscid and incompressible flow, the components of the velocity of the flow and the pressure distributions on the rockets may be derived from the spatial variation of the stream function. The spatial variation of the stream function is primarily controlled by the requirement that Laplace's equation must be satisfied at every point in the flow field; that is,

$$\nabla^2 \psi = 0 \ . \tag{3}$$

Those lines in the flow field along which the stream function has a constant value are known as streamlines. The physical significance of the stream function may be considered to be that the volume rate of flow which passes between streamlines is constant.

Where complexities in the flow field prohibit the determination of analytical solutions to Eq. 3, numerical methods may be employed to obtain solutions to specific problems. In these numerical methods the requirement that the controlling equation be satisfied at every point in the flow field is replaced by the less stringent one that a finite difference approximation to the controlling equation must be satisfied only at discrete points in the flow field. The location of these points is determined by superimposing a rectangular mesh or network of lines upon the flow field and taking the intersection of these lines as the discrete or nodal points at which the finite difference equations are applied.

In the present work a square grid has been selected and used with a five point finite difference approximation to Laplace's equation. A sketch of a regular mesh point say, for example, point 0 , is shown in Figure 3. Such a point is termed regular because all the lines radiating

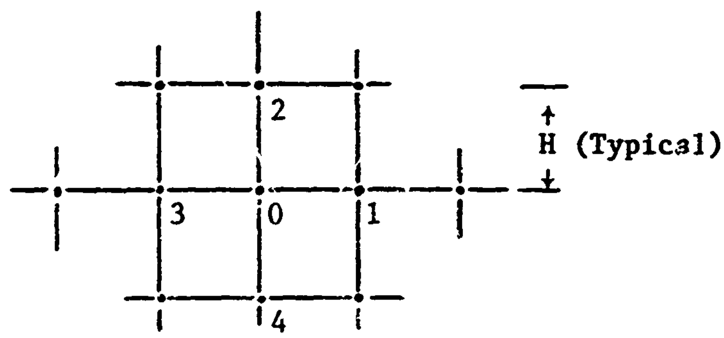

FIGURE 3. A REGULAR MESH POINT AND ITS SURROUNDINGS

from it terminate on points located a distance H away. In this case the approximation to Laplace's equation may be shown to be,

$$\psi_0 = \frac{1}{4} (\psi_1 + \psi_2 + \psi_3 + \psi_4) \ . \tag{4}$$

The subscripts correspond to those nodal points from which the values of the stream function are taken. Although the equations for the irregular points that lie adjacent to solid bodies will be somewhat complicated by one or more of the arms having lengths less than H , the value of the stream function at a point remains a function of the values of the stream function which bound the point. Equations similar to Eq. (4) may be written for every nodal point in the flow field. The simultaneous solution of all of these nodal point equations yields the spatial variation of the stream function in the flow field.

From the many methods available for the computation of the solution values of the stream function at the nodal points, an iterative type, point by point technique has been selected. In this method, initial values of the stream function are arbitrarily assigned to all the nodal points that lie inside the flow field. Computation then proceeds, row after row in succession, until values at each nodal point in the network have been recalculated. This procedure is repeated until successive iterations through the complete network result in fractional changes in the stream function at every nodal point that are less than the value of TOL specified in the input data. Upon the completion of two such iterations, the values at the nodal points are considered to be close enough to the values that satisfy the simultaneous finite difference equations and the spatial variation of the stream function is established.

During the n^{th} iteration through the flow field, the value of the stream function at a particular point depends upon values of the stream function at surrounding points that are improved and unimproved; that is, the values are the result of calculations performed during both the current, n^{th} , and preceding, $n-1^{th}$, iterations. If superscripts in parentheses are used to indicate the number of the iteration in which the value of the stream function was calculated, then, considering point 0 in Fig. 3, the technique of point by point computation is illustrated by the following equation:

$$\psi_0^{(n)} = \frac{1}{4}\left[\psi_1^{(n-1)} + \psi_2^{(n)} + \psi_3^{(n)} + \psi_4^{(n-1)}\right] \tag{5}$$

The iterative technique indicated by Eq. (5) is known as the Gauss-Seidel method and has been used frequently and successfully. However, the number of iterations required to bring the changes in the stream function within tolerable limits may be substantially reduced by the incorporation of a convergence or relaxation factor. When such a factor is added, the technique is known as the method of successive overrelaxations (4). The equation for the stream function at point 0 as computed by the method of successive overrelaxations is, during the n^{th} iteration:

$$\psi_0^{(n)} = W\left\{\frac{1}{4}\left[\psi_1^{(n-1)} + \psi_3^{(n)} + \psi_4^{(n-1)}\right]\right\} - (W-1)\psi_0^{(n-1)} \tag{6}$$

It may be observed that when the value of W is taken as one, the method of successive overrelaxations reduces to the Gauss-Seidel method. However, when the optimum value of W is used, it has been shown (4) that the method of successive overrelaxations converges approximately $2/\pi H$ times as rapidly as the Gauss-Seidel method and it was for this reason that the method of successive overrelaxations was selected. The optimum value of W for a rectangular flow field depends on the number of mesh points comprising the length and width of the field. Equations for the estimation of this value are given in the section of this report that describes the input data to the program. Although the flow field is indeed rectangular, it is also pierced in two places by the rockets so that the equations for optimum values of W are not exact. The degree of approximation which is involved may be established by examining the variation of the number of iterations required to complete a particular problem with the value of W. A guideline that may be useful in carrying out this investigation is that the number of iterations increases much more rapidly when W is less than its optimum value than when W is overestimated.

The way in which the velocity depends upon the spatial variation of the stream function may, perhaps, best be illustrated by a discussion of the very important problems associated with the boundary conditions. The specification of the values of the stream function on the boundary of the flow field and the equation controlling the behavior of the stream function inside the region completely determine the solution to a problem of this type. Although the mechanics of determining the implied solution may be tedious, demanding, and, in some cases, impossible, the solution is nonetheless fixed by a statement of the boundary conditions and the controlling equation. It is for this reason that the boundary conditions are considered to be so important. In the present case the values of the stream function on the upstream boundary and the boundaries lying parallel to the direction of flow may be found without difficulty; however, the velocity distribution or the variation of the stream function on the downstream boundary cannot be specified with any degree of confidence. It does appear that the influence of the conditions at the rear boundary on the forces of interaction may be minimized by placing the rear boundary a considerable distance downstream from the rockets, but this must be checked by computing several cases with the program.

On the upstream boundary the input values of the stream function are easily determined, provided the boundary is located sufficiently far from the nose of the rocket that the flow field may be assumed to be undisturbed. If u is taken to be the component of the velocity in the x direction and v as the velocity component in the y direction, both are in general functions of position,

$$u = u(x,y), \tag{7}$$

$$v = v(x,y). \tag{8}$$

They are related to the stream function through the equations,

$$u = -\frac{\partial\psi}{\partial y} \tag{9}$$

and

$$v = \frac{\partial\psi}{\partial x} \quad . \tag{10}$$

A diagram indicating a coordinate system and the indexing technique employed in the program is presented in Figure 4.

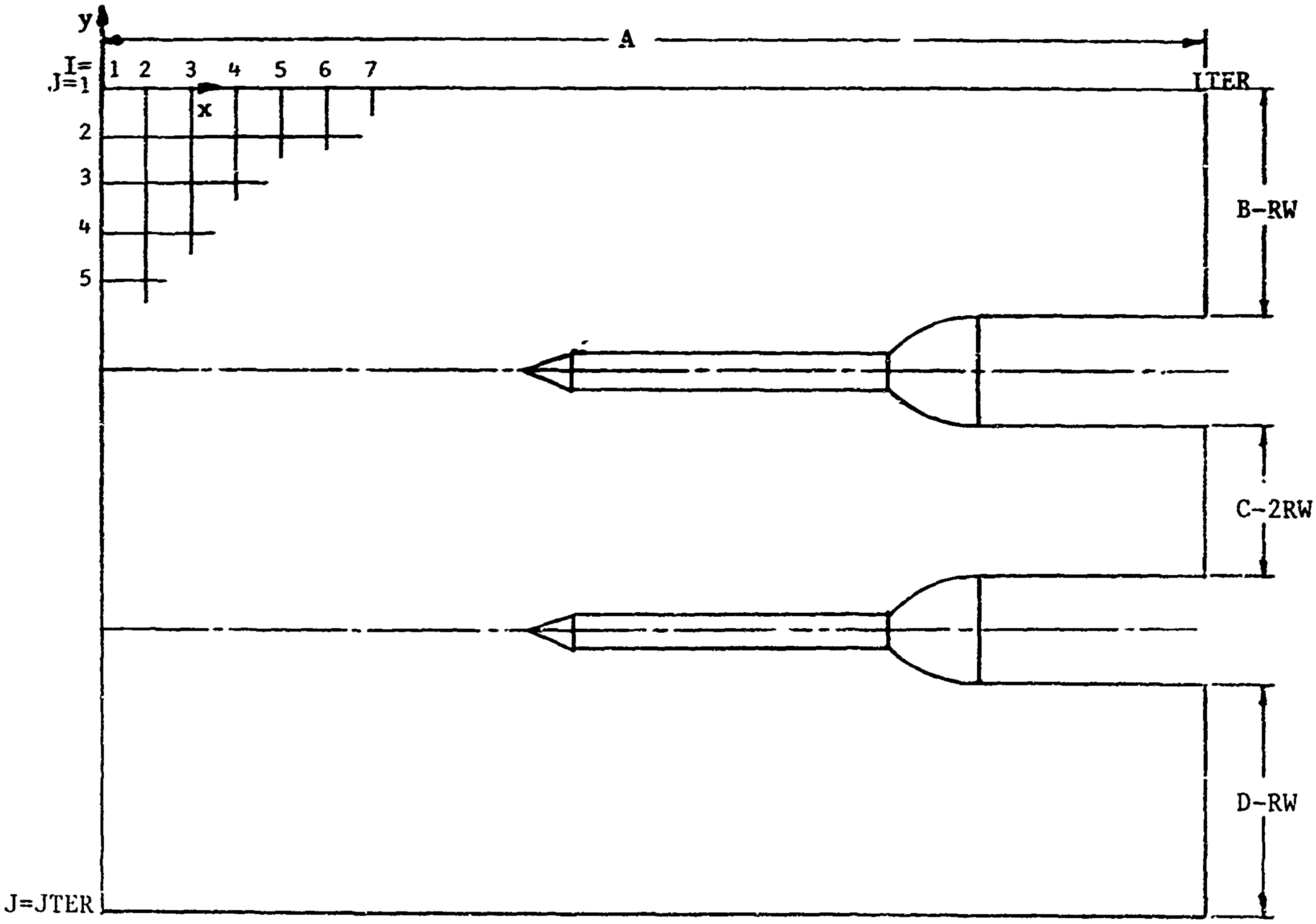

FIGURE 4. DETERMINATION OF THE STREAM FUNCTION ON THE BOUNDARY OF THE FLOW FIELD

In the far-upstream region the velocity is constant and equal to the free stream velocity. Along the upstream boundary then the components of the velocity are:

$$u(0, y) = U_\infty \text{ , the free stream velocity,} \tag{11}$$

and

$$v(0, y) = 0 \ . \tag{12}$$

At any point in the flow field, the stream function is related to the components of the velocity through the partial differential equation,

$$d\Psi = v(x,y)dx - u(x,y)dy \, . \tag{13}$$

Integrating this equation along the line forming a part of the upstream boundary and passing between the origin and a point $y = -Y$ from the origin yields an expression for the variation of the stream function along this boundary,

$$\Psi(0,Y) = \Psi(0,0) + U_\infty Y \ . \tag{14}$$

The value of $\Psi(0,0)$ in the above equation is completely arbitrary. This equation may now be translated into the finite difference terminology used in the program. The distance Y to the nodal point located on the column $I = 1$ and some row J is given by

$$Y = H(J - 1) \ . \tag{15}$$

Where values of the stream function are denoted $X(I,J)$ and the values of I and J inside the parentheses denote the nodal point with which the value of the stream function is associated, Eq. (14) may be rewritten:

$$X(1,J) = X(1,1) + (J-1)HU_\infty \, . \tag{16}$$

The arbitrary value $\Psi(0,0)$ in Eq. (14) is now identified as $X(1,1)$ in Eq. (16).

Values of the stream function on the boundaries parallel to the flow direction are also implied by the above equation. The value specified in the input data for that point on the upstream boundary which is common to both the upstream and a longitudinal boundary is taken by the program as the value of the stream function at every nodal point on that longitudinal boundary. In equation form,

$$X(1,1) = X(2,1) = X(3,1) = \ldots = X(ITER, 1) \quad (17)$$

and

$$X(1,JTER) = X(2,JTER) = X(3,JTER) = \ldots = X(ITER,JTER) \quad (18)$$

This method for the specification of the stream function on the longitudinal boundaries assumes the boundaries are streamlines and that no fluid passes across them. Such an assumption is justified when the longitudinal boundaries are placed at such a far distance from the rockets that the fluid accelerated normal to the direction of flight of the rockets exits from the flow field through the rear boundary without reaching the longitudinal boundary.

The values of the stream function that are specified on the upstream boundary are used by the program in yet another way. The initial values of the stream function at points inside the flow field are obtained by setting the stream function at all interior points on a particular row equal to the value specified in the input data at the point of intersection of that row and the upstream boundary. For row number J the initially assumed values are given by,

$$X(1,J) = X(2,J) = X(3,J) = \ldots = X(ITER-1,J) \quad (19)$$

where J equals neither JTER nor one. Although these values are arbitrary in that they do not affect the final answer, the number of iterations required for the program to converge to the final answer may be significantly affected. That is, the more nearly the initially assumed values of the stream function correspond to their final distribution the fewer will be the number of iterations that are required. As experience is gained in the problem, it may appear that improvements can be made in the method for assuming the initial distribution in the stream function. The incorporation of such an improvement should result in substantial savings in computer time.

Along the downstream boundary the assumption of a uniform velocity is obviously untenable and experimental data are not available on velocity distributions in the vicinity of two exhausting jets in a uniform flow stream. Although the program developed here can shed little light upon the velocity distributions that actually occur in such geometries, it can be used to assess, insofar as the interacting forces are concerned, the relative importance of this gap in our knowledge. By examining the effect of several velocity distributions placed at various distances downstream from the rockets, the extent of the influence of this variable may be established. It often happens in this kind of problem that the influence of the conditions existent on a particular boundary may be minimized by moving the boundary a sufficient distance away from the point of interest. However, in the event the variation of the stream function on the downstream boundary is shown to be a significant variable and one whose influence cannot be reduced below a tolerable level, the program should be supplemented with experimental investigations.

Several criteria which the velocity distribution on the rear boundary must fulfill may be derived from theoretical considerations. Returning to the controlling partial differential equation, Eq. (13), and performing another line integration along $x = A$ from the point $y = 0$ downward a distance Y, the relation between the stream function and the velocity distribution is found to be:

$$\Psi(A,-Y) - \Psi(A,0) = - \int_{y=0}^{y=-Y} u(A,y)dy \ , \qquad (20)$$

where the distance Y is less than $B-RW$. Where Y equals $B-RW$, Eq. (20) becomes, in terms of the relaxation variables,

$$XB1 = X(1,1) - \int_{0}^{-B+RW} u(A,y)dy \ . \qquad (21)$$

The variable $XB1$ is the value of the stream function at every point on the boundary of the leading rocket and, also, it is the value of the streamline that intersects the apex of the nose cone of the leading rocket. A similar relation may be derived for the portion of the downstream boundary that lies between the two rockets:

$$\Psi(A,-Y) - \Psi(A,-B-RW) = - \int_{y=-B-RW}^{y=-Y} u(A,y)dy \ , \qquad (22)$$

which, translating to relaxation variables and letting Y go to $B + C - RW$, yields,

$$XB2 - XB1 = - \int_{y=-B-RW}^{y=-B-C+RW} u(A,y)dy \ . \qquad (23)$$

The variable $XB2$ plays the same role in the case of the trailing rocket as does $XB1$ for the leading rocket. Finally, for the portion of the downstream boundary that remains, we have:

$$\psi(A,-Y) - \Psi(A,-B-C-RW) = -\int_{y=-B-C-RW}^{y=-Y} u(A,y)dy \tag{24}$$

and over the entire length of that portion,

$$X(1,JTER) - XB2 = -\int_{y=-B-C-RW}^{y=-B-C-D} u(A,y)dy \ . \tag{25}$$

These equations relating the velocity distribution, u(A,y), to the stream function along the downstream boundary should be of some assistance in determining the importance of the flow conditions specified at the downstream boundary.

The Eqs. (21), (23) and (25) do not completely define the values of XB1 and XB2 . These variables are also subject to speculation. When the two rockets fly with zero lag of one behind the other, the values of XB1 and XB2 should be equal to the values of the stream function that were specified on the upstream boundary at the intersection of the upstream boundary and the longitudinal axes of the respective rockets. Physically, this means that flows initially directed between the two rockets continue that orientation without deflection. When one rocket trails the other the value to be assigned to XB2 is subject to question. The leading rocket deflects some of the flow so that a part of the fluid that was initially directed to pass between the two rockets will pass between the trailing rocket and the boundary of the flow field nearest to it. Data are not available on the fraction of the flow that is deflected and the present computation will not yield any such information. However, the program may, again, be employed to establish the relative importance of this variable and provide an estimation of the magnitude of its influence on the interacting forces.

Once the spatial variation of the stream function has been established to within a satisfactory degree of accuracy, the relaxation phase of the program is complete. In the next step the net aerodynamic force and center of pressure are computed for both rockets. A sketch of the lead rocket is shown in Figure 5; this figure will be used to illustrate the derivation of the equations employed in the final computational phase of the program.

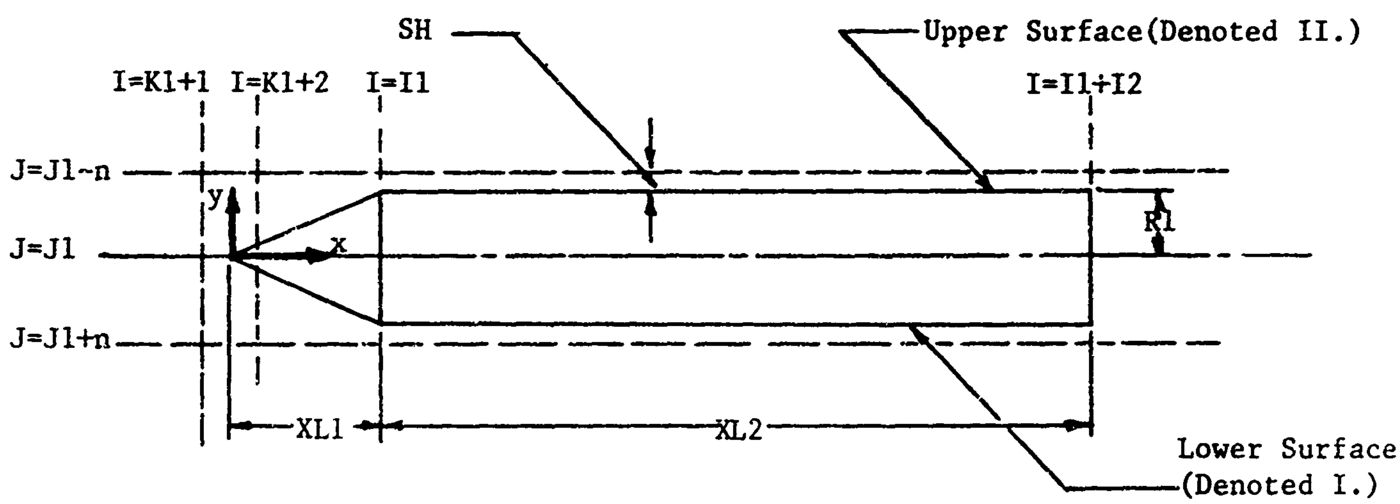

FIGURE 5. COMPUTATION OF AERODYNAMIC FORCE AND CENTER OF PRESSURE ON THE LEADING ROCKET

Those variables that are to be evaluated at the upper surface will be denoted by the subscript II while those that apply to the lower surface will be subscripted I . Several of the more important mesh lines are dashed in on the figure and given names for ready reference. The value of the variable n is an integer such that the mesh lines J1 + n and J1 - n lie closest to and parallel with the cylindrical portion of the rocket.

Taking the upward direction as positive, the net force of this rocket may be computed by evaluating the following integrals,

$$F = \int_{x=0}^{x=XL1} (p_I - p_{II})dx + \int_{x=XL1}^{x=XL2+XL1} (p_I - p_{II})dx \ , \qquad (26)$$

where p stands for the pressure in the fluid. The pressure variable may be eliminated from Eq. (26) by the use of Bernoulli's equation,

$$\frac{p}{\rho_\infty} + \frac{q^2}{2} = \frac{p_\infty{}^2}{\rho_\infty} + \frac{U_\infty{}^2}{2} \ , \qquad (27)$$

where ρ_∞ is the density of the atmosphere and q is the magnitude of the velocity vector. Substitution yields,

$$F = \frac{\rho_\infty}{2} \left\{ \int_{x=0}^{x=XL1} (q_{II}^2 - q_I^2)dx + \int_{x=XL1}^{x=XL1+XL2} (q_{II}^2 - q_I^2)dx \right\} . \qquad (28)$$

The magnitude of the velocity vector is equal to the sum of the squares of the velocity components which are in turn related to the spatial derivatives of the stream function:

$$q^2 = u^2 + v^2 = \left(\frac{\partial\psi}{\partial y}\right)^2 + \left(\frac{\partial\psi}{\partial x}\right)^2 \ . \qquad (29)$$

Substituting these values into the integral for F and realizing that $\frac{\partial\psi_I}{\partial x} = \frac{\partial\psi_{II}}{\partial x} = 0$ on the boundaries of the rocket between $x = XL1$ and $x = XL1+XL2$, yields:

$$F = \frac{\rho_\infty}{2}\int_{x=0}^{x=XL1} \{(\frac{\partial\psi_{II}}{\partial y})^2 + (\frac{\partial\psi_{II}}{\partial x})^2 - (\frac{\partial\psi_I}{\partial y})^2 - (\frac{\partial\psi_I}{\partial x})^2 dx + \frac{\rho_\infty}{2}\int_{x=XL1}^{x=XL1+XL2} \{(\frac{\partial\psi_{II}}{\partial y})^2 + (\frac{\partial\psi_I}{\partial y})^2\}dx. \quad (30)$$

In writing the finite difference approximation to these integrals it will be simpler if the following additional definitions are introduced:

$$\bar{J} = J1 + n, \quad (31)$$

$$\tilde{J} = J1 - n. \quad (32)$$

As an example consider the evaluation of the last integral in Eq. (30). The partial derivatives may be approximated along any mesh column between I1 and I1+I2 by the following two point analogies:

$$\frac{\partial\psi_I}{\partial y} \simeq \frac{XB1-X(I,\bar{J})}{SH} \quad (33)$$

and

$$\frac{\partial\psi_{II}}{\partial y} \simeq \frac{X(I,\tilde{J})-XB1}{SH}. \quad (34)$$

As is conventional in this type computation, the integration is replaced by a summation. The approximation to the integral may be shown to be equivalent to,

$$\int_{x=XL1}^{x=XL1+XL2} \{(\frac{\partial\psi_{II}}{\partial y})^2 + (\frac{\partial\psi_I}{\partial y})^2\}dx = \frac{H}{(SH)^2}\sum_{I=I1}^{I=I1+I2} G(I)\{[X(I,\tilde{J})-XB1]^2 - [XB1-X(I,\bar{J})]^2\} \quad (35)$$

where,

$$G(I) = \begin{cases} 0.5 \text{ @ } I=I1 \\ 1.5 \text{ @ } I=I1+1,\ I1+2,\ \ldots,\ I1+I2-1 \\ 0.5 \text{ @ } I=I1+I2. \end{cases}$$

The finite difference approximation to the remaining integral in Eq. (30) is somewhat more complicated because the surface is not horizontal. Breaking it into the sum of two integrals,

$$\int_{x=0}^{x=XL1} \{(\frac{\partial\psi_{II}}{\partial y})^2 + (\frac{\partial\psi_{II}}{\partial x})^2 - (\frac{\partial\psi_{I}}{\partial y})^2 - (\frac{\partial\psi_{I}}{\partial x})^2\}dx =$$

$$\int_{x=0}^{x=XL1} \{(\frac{\partial\psi_{II}}{\partial y})^2 - (\frac{\partial\psi_{I}}{\partial y})^2\}dx + \int_{x=0}^{x=XL1} \{(\frac{\partial\psi_{II}}{\partial x})^2 - (\frac{\partial\psi_{I}}{\partial x})^2\}dx. \qquad (36)$$

Then the finite difference approximation to the first integral on the right hand side of Eq. (36) is:

$$\int_{x=0}^{x=XL1} \{(\frac{\partial\psi_{II}}{\partial y})^2 - (\frac{\partial\psi_{I}}{\partial y})^2\}dx = \sum_{I=K1+2}^{I=I1} \bar{G}(I) \left\{\frac{[X(I,\bar{J})-XB1]^2 - [XB1-X(I,\bar{J})]^2}{[H\cdot(\bar{J}-J1) - \frac{ARM\cdot R1}{XL1}]^2}\right\} \qquad (37)$$

where,

$$\bar{G}(I) = \begin{cases} 0.5 + ARM & @\ I = K1+2 \\ H & @\ K1+3 \le I \le I1-1 \\ 0.5H & @\ I = I1 \text{ and } I1+2 \le I1 \\ XL1 & @\ I = I1 \text{ and } I1+2 \ge I1 \end{cases}$$

and $ARM = XL1-H\cdot(I1-I)$.

For the second integral on the right hand side of Eq. (36),

$$\int_{x=0}^{x=XL1} \{(\frac{\partial\psi_{II}}{\partial x})^2 + (\frac{\partial\psi_{I}}{\partial x})^2\}dx = \sum_{J=1}^{J=n-1} H\cdot\left\{\frac{[XB1-X(\bar{I},J1-J)]^2 - [XB1-X(\bar{I},J1+J)]^2}{(SH)^2}\right\}$$

where, $SH = H\cdot(I1-\bar{I}) - \{L1-H\cdot(J)\cdot\frac{XL1}{R1}\}$. (38)

Evaluation of these integrals yields the approximate value of the net upward force acting on the leading rocket. The center of pressure is calculated in a parallel computation which is the evaluation of,

$$\bar{x} = \frac{1}{F}\left\{\int_{x=0}^{x=XL1} (p_I - p_{II})x\, dx + \int_{x=XL1}^{x=XL1+XL2} (p_I - p_{II})x\, dx\right\}. \qquad (39)$$

A similar set of equations has been developed for the net upward force and center of pressure on the trailing rocket.

CONCLUSION

A computer program has been developed which will compute the mutual interaction between a pair of rockets fired essentially simultaneously. Preliminary computations performed with this program indicate a sizable aerodynamic interaction does occur and could well explain the observed behavior of rockets in simultaneous or salvo firings.

As has been indicated in the preceding discussion, the influence of a number of factors must be determined before the computation method can be considered a reliable one. Such variables as the velocity distribution on the downstream boundary, the value of the streamline intercepting the apex of the nose cone of the trailing rocket, the distance the boundary must be placed away from the rockets in order that the flow may be considered to be undisturbed, are the major problems that require attention. The results of an investigation of these variables may indicate that the program will satisfactorily predict the interaction between rockets in the velocity range for which it is designed, but it should be realized that the possibility exists that experimental work may be required.

In addition to the theoretical considerations there are others that have to do with optimizing the computing time. The value of H and TOL must be sufficiently small that they affect neither the force nor the center of pressure on the rockets, but they must also be as large as is practical in order to minimize running time and remain within memory capacity. The optimum values of these variables can be established only by computation with the program.

When these investigations have been completed, the influence of such variables as rocket velocity, spacing between rockets, and distance of lag of one behind the other on the force of interaction may be investigated.

BIBLIOGRAPHY

1. S.-I. Pai, "Fluid Dynamics of Jets," D. Van Nostrand Company, Inc., New York, 1954, Chapters V, VI and VII .

2. E. S. Love, C. E. Grigsby, L. P. Lee, and M. J. Woodling, "Experimental and Theoretical Studies of Axisymmetric Free Jets," NASA TR R-6, 1959.

3. Milne - Thomson, "Theoretical Hydrodynamics," MacMillan Company, New York, 3rd Ed., 1955, Chapters V and VI.

4. John Todd, "A Survey of Numerical Analysis," McGraw-Hill Book Co., New York, 1962, Chapter 11.